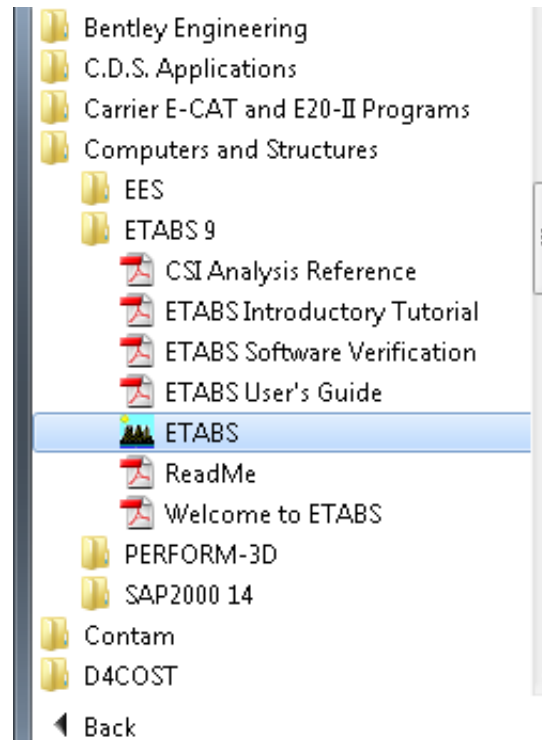


Quick Typical Bay Design in ETABS

-This guide will show you how to get a ballpark estimate of beam and column sizes for a typical bay in your structure

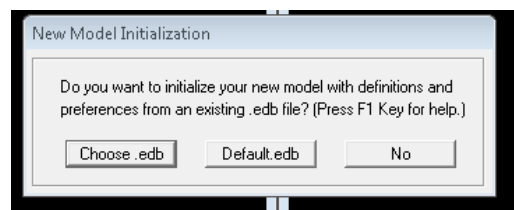
1. Open ETABS

Go to All Programs---Computers and Structures Folder---ETABS Folder---ETABS



2. Set up New Model

Select File---New Model---Answer No to the question asked



3. Edit Building Plan Grid System and Story Data Definition

Building Plan Grid System and Story Data Definition

Grid Dimensions (Plan)

☒ Uniform Grid Spacing

Number Lines in X Direction: 4

Number Lines in Y Direction: 4

Spacing in X Direction: 288

Spacing in Y Direction: 288

☐ Custom Grid Spacing

Grid Labels... Edit Grid...

Story Dimensions

☒ Simple Story Data

Number of Stories: 4

Typical Story Height: 144

Bottom Story Height: 144

☐ Custom Story Data Edit Story Data...

Units

Kip-in

Add Structural Objects

Steel Deck Staggered Truss Flat Slab Flat Slab with Perimeter Beams Waffle Slab Two Way or Ribbed Slab **Grid Only**

OK Cancel

Note the units selected.

For this example, I will just be doing a one story bay. Input as follows:

Building Plan Grid System and Story Data Definition

Grid Dimensions (Plan)

☒ Uniform Grid Spacing

Number Lines in X Direction: 2

Number Lines in Y Direction: 2

Spacing in X Direction: 27

Spacing in Y Direction: 30

☐ Custom Grid Spacing

Grid Labels... Edit Grid...

Story Dimensions

☒ Simple Story Data

Number of Stories: 1

Typical Story Height: 12

Bottom Story Height: 12

☐ Custom Story Data Edit Story Data...

Units

Kip-ft

Add Structural Objects

Steel Deck Staggered Truss Flat Slab Flat Slab with Perimeter Beams Waffle Slab Two Way or Ribbed Slab **Grid Only**

OK Cancel

4. You can then select the Structural System that you are designing. This example will show you how to do a Steel System with Steel Deck.

After editing the dimension values, click steel deck.

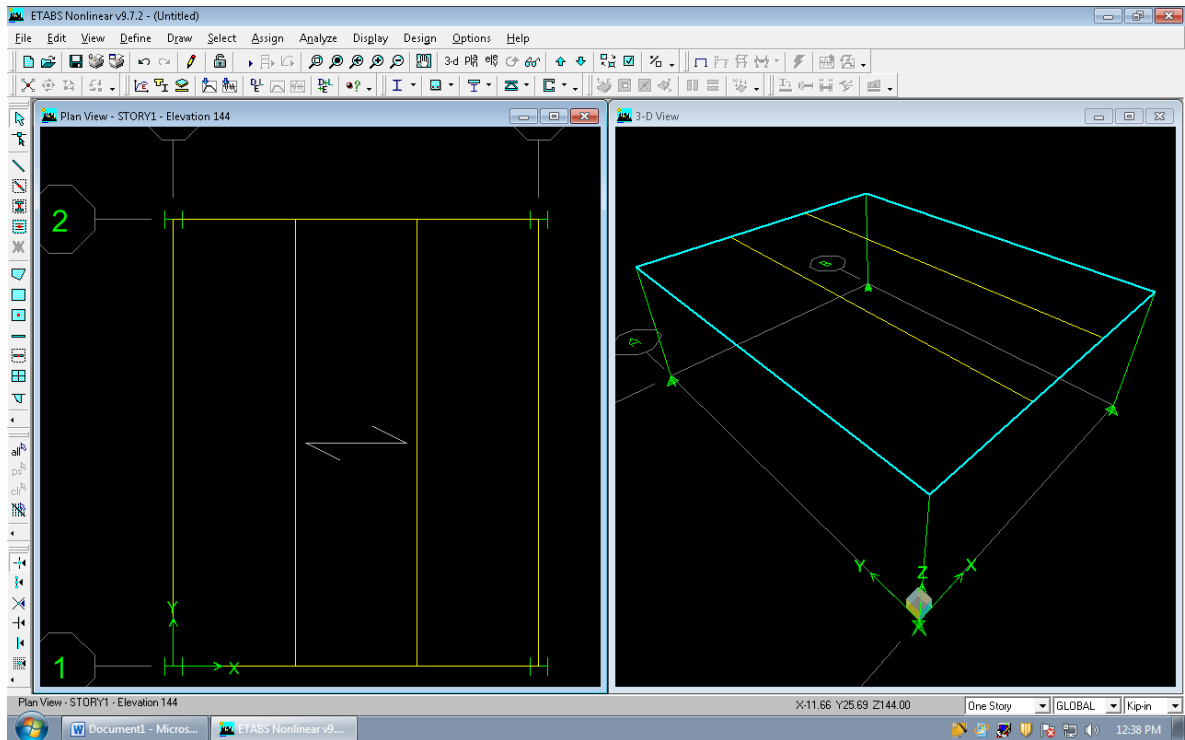
The screenshot shows the 'Steel Floor System' dialog box. The 'Overhangs' section has 'Along X Direction' with 'Left Edge Distance' at 0.5 and 'Right Edge Distance' at 0.5, and 'Along Y Direction' with 'Top Edge Distance' at 0.5 and 'Bottom Edge Distance' at 0.5. The 'Secondary Beams' section has 'Secondary Beams' checked, 'Direction' set to 'X', and 'Number' set to 3. The 'Moment Frame Type' section has 'Intersecting' selected. The 'Restrains at Bottom' section has 'Pinned' selected. The 'Structural System Properties' section has 'Lateral Column' as 'A-LatCol', 'Lateral Beam' as 'A-LatBm', 'Gravity Column' as 'A-GravCol', 'Gravity Beam' as 'A-GravBm', 'Secondary Beam' as 'A-CompBm', and 'Deck/Floor' as 'DECK1'. The 'Load' section has 'Dead Load Case' as 'DEAD', 'Dead Load (Additional)' as 0, 'Live Load Case' as 'LIVE', and 'Live Load' as 0. The 'Create Rigid Floor Diaphragm' checkbox is checked. A small diagram of a floor bay is shown in the center, and 'OK' and 'Cancel' buttons are at the bottom right.

You can now edit the floor system to your typical bay.

The screenshot shows the 'Steel Floor System' dialog box with edited settings. The 'Overhangs' section has 'Along X Direction' with 'Left Edge Distance' at 0 and 'Right Edge Distance' at 0, and 'Along Y Direction' with 'Top Edge Distance' at 0 and 'Bottom Edge Distance' at 0. The 'Secondary Beams' section has 'Secondary Beams' checked, 'Direction' set to 'Y', and 'Number' set to 2. The 'Moment Frame Type' section has 'Intersecting' selected. The 'Restrains at Bottom' section has 'Pinned' selected. The 'Structural System Properties' section has 'Lateral Column' as 'A-LatCol', 'Lateral Beam' as 'A-LatBm', 'Gravity Column' as 'A-GravCol', 'Gravity Beam' as 'A-GravBm', 'Secondary Beam' as 'A-CompBm', and 'Deck/Floor' as 'DECK1'. The 'Load' section has 'Dead Load Case' as 'DEAD', 'Dead Load (Additional)' as 0, 'Live Load Case' as 'LIVE', and 'Live Load' as 0. The 'Create Rigid Floor Diaphragm' checkbox is checked. A small diagram of a floor bay is shown in the center, and 'OK' and 'Cancel' buttons are at the bottom right.

Note: You can specify Superimposed Dead Load and Live Load in this step or later. I chose to specify it later.

Then Click Ok and then Ok again and you're bay should appear on screen similar to below.



5. Edit your Deck Dimensions

Go to Select---by Area Object Type---Floor---Click Ok
Your floor deck should be selected

Next, Go to Assign---Shell/Area---Wall/Slab/Deck Section---Choose Deck 1---Modify
Section

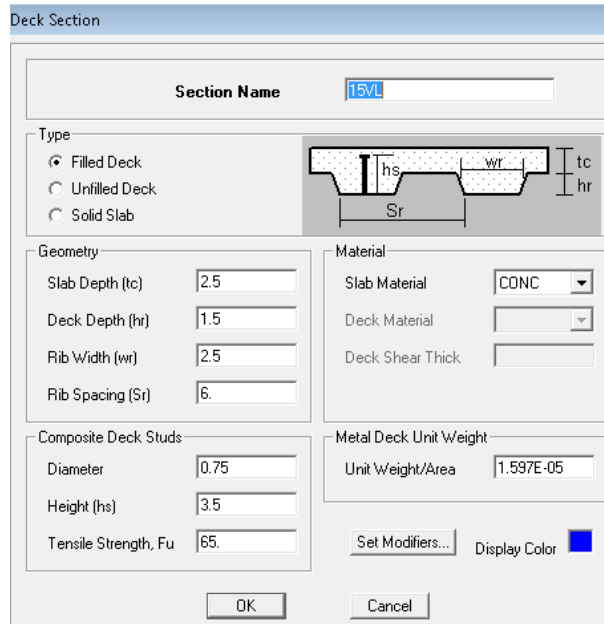
The window below should appear.

The 'Deck Section' dialog box is shown with the following settings:

- Section Name:** DECK1
- Type:** ☒ Filled Deck, ☐ Unfilled Deck, ☐ Solid Slab
- Geometry:**
 - Slab Depth (tc): 3.5
 - Deck Depth (hr): 3
 - Rib Width (wr): 6
 - Rib Spacing (Sr): 12
- Material:**
 - Slab Material: CONC
 - Deck Material: (empty)
 - Deck Shear Thick: (empty)
- Composite Deck Studs:**
 - Diameter: 0.75
 - Height (hs): 6
 - Tensile Strength, Fu: 65
- Metal Deck Unit Weight:**
 - Unit Weight/Area: 1.597E-05
- Buttons:** Set Modifiers..., Display Color (checked), OK, Cancel

Edit the information to correspond with the floor deck that you have designed.

For example,



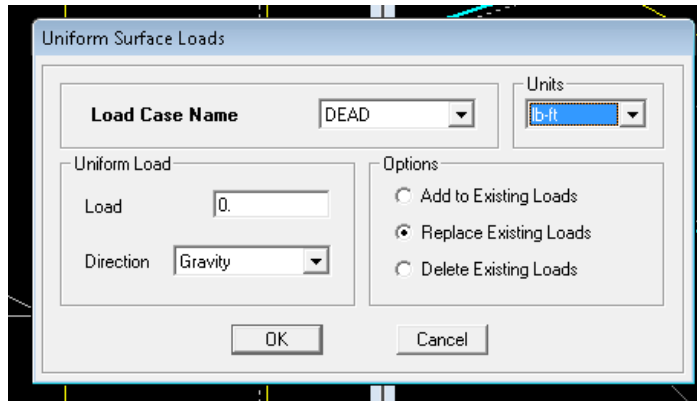
6. Place Loads on the Structure

Go to Select---by Area Object Type---Floor---Click Ok

Next, Go to Assign---Shell/Area Loads---Uniform

Here you can add Superimposed Dead Loads as well as Live Loads

Note the units you select here so that you input the correct load (typically loads are in psf)



Assign as many loads as you want to apply following the same procedure

Note the Load Case you want to place the load in and whether you want to add to existing loads, replace existing loads, or delete the existing loads in that load case.

You may also add lateral loads, but for this example I will only consider gravity loads

7. Run Analysis

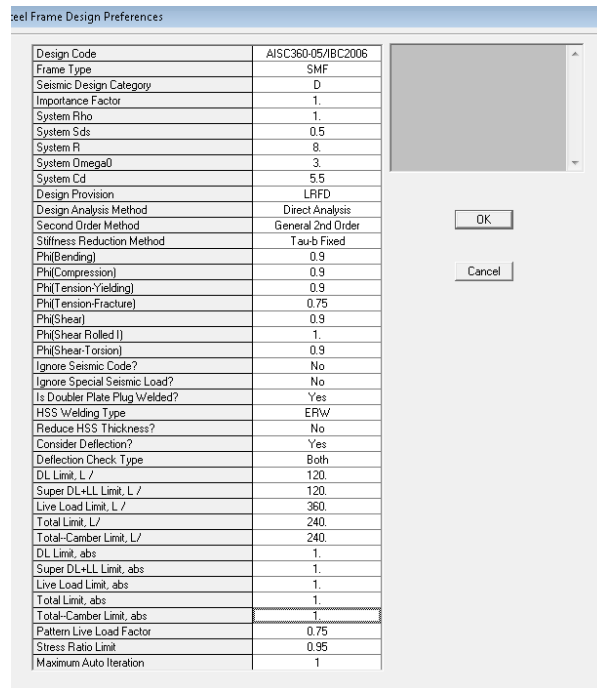
Go to Analyze---Run Analysis

Now, we are ready for the design procedure.

8. Set Design Options

Go to Options---Preferences---Steel Frame Design

Now you can set design parameters and the design code you would like to use.



The image shows a software dialog box titled "Steel Frame Design Preferences". It contains a table of design parameters and their values, along with "OK" and "Cancel" buttons.

Design Code	AISC360-05/IBC2006
Frame Type	SMF
Seismic Design Category	D
Importance Factor	1.
System Rho	1.
System Sds	0.5
System Ri	8.
System Omega0	3.
System Cd	5.5
Design Provision	LRFD
Design Analysis Method	Direct Analysis
Second Order Method	General 2nd Order
Stiffness Reduction Method	Tau-b Fixed
Phi(Bending)	0.9
Phi(Compression)	0.9
Phi(Tension-Yielding)	0.9
Phi(Tension-Fracture)	0.75
Phi(Shear)	0.9
Phi(Shear-Rolled I)	1.
Phi(Shear-Torsion)	0.9
Ignore Seismic Code?	No
Ignore Special Seismic Load?	No
Is Doubler Plate Plug Welded?	Yes
HSS Welding Type	ERW
Reduce HSS Thickness?	No
Consider Deflection?	Yes
Deflection Check Type	Both
DL Limit, L /	120
Super DL+LL Limit, L /	120
Live Load Limit, L /	360
Total Limit, L /	240
Total-Camber Limit, L /	240
DL Limit, abs	1.
Super DL+LL Limit, abs	1.
Live Load Limit, abs	1.
Total Limit, abs	1.
Total-Camber Limit, abs	1.
Pattern Live Load Factor	0.75
Stress Ratio Limit	0.95
Maximum Auto Iteration	1

Also set the design options for a composite steel design following the same procedure.

Preferences

Factors	Deflection	Vibration	Price
Beam			

Shored?	No
Middle Range (%)	70.
Pattern Live Load Factor	0.75
Stress Ratio Limit	1.
Single Segment for Studs?	No
Stud Increase Factor	1.
Additional Minimum Studs	0

Design Code AISC360-05/IBC ▾

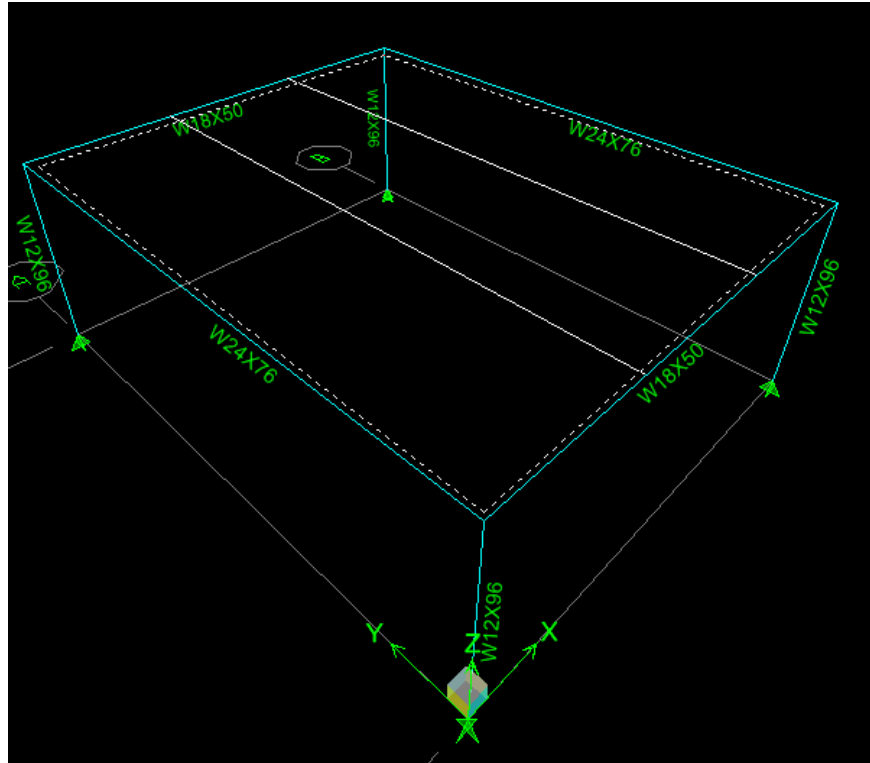
Note you can remove camber options in the deflection tab

By doing this, you can compare design results accurately between the two systems.

9. Design Structure

Go to Design---Steel Frame Design---Start Design/Check of System

The sizes will now appear on the structure like below



You can then follow the same procedure to do a composite steel design and compare your results

Conclusion

You now have a basic range of sizes to expect when designing your structure

It is recommended that you do calculations by hand or by using Excel, to ensure your design satisfies code requirements.